International Journal of Engineering Studies.
ISSN 0975-6469 Volume 8, Number 2 (2016), pp. 129-145

http://www.ripublication.com

Study on Sand Particles Drying in a Fluidized Bed Dryer using CFD

Jibin Antony[1] **and M.B. Shyamkumar**[2]

[1]M.Tech Student, School of Mechanical and Building Sciences,
VIT University, Chennai-600 127, India.
Email: jvath.007@gmail.com

[2]Associate Professor, School of Mechanical and Building Sciences,
VIT University, Chennai-600 127, India.
Email: shyamkumar.mb@vit.ac.in

Abstract

Drying of sand particles is one of the important operations in a mining and mineral separation plant. For drying mineral rich sand particles from the captive beaches, mainly the Fluidized Bed Dryers (FBD) are employed, which offers many advantages over other drying methods. In the present study, drying of sand particles using a Fluidized Bed Dryer was simulated using ANSYS FLUENT 14.5. The simulations are based on the Eulerian-Eulerian multiphase model approach. The contours of volume fraction of sand over time at a mid-plane inside the FBD are monitored at an inlet air temperature of 400 K. The simulations are further carried out for different inlet air temperatures. As inlet air temperature increases, the drying time is reduced. The simulated results namely the outlet air temperature and velocity are found to be in good agreement with those of the experimental data obtained from the Mineral Separation (MS) Unit of Kerala Minerals and Metals Limited (KMML), Kollam.

Keywords: Fluidized bed dryer, Multiphase fluid flow, Eulerian–Eulerian model, Volume fraction, Mineral sand.

INTRODUCTION

Fluidized bed dryers are common for drying of wet sand particles, pellets, powders and other chemicals in various industries such as mineral separation plant, petroleum, agricultural and pharmaceutical industries. The process involved in drying using fluidized bed dryer is convection where the heated air in direct contact with the wet feed, dries the material suspended in a fluidizing bed. In the drying process, the air from a blower is heated by a natural gas fueled air heater and passes upward into the dryer through a perforated air distributor plate. This fluidizes the wet feed material. As the heated air passes through the wet material, the air carries away the moisture content and is exhausted through an outlet. Fig.1 shows the drying process of sand particles. The advantages of fluidized bed dryers over other drying methods are (1) high drying rate because of good interaction between the particles and air, (2) better temperature control and operation throughout the process and (3) less maintenance cost due to absence of moving components inside the dryer.

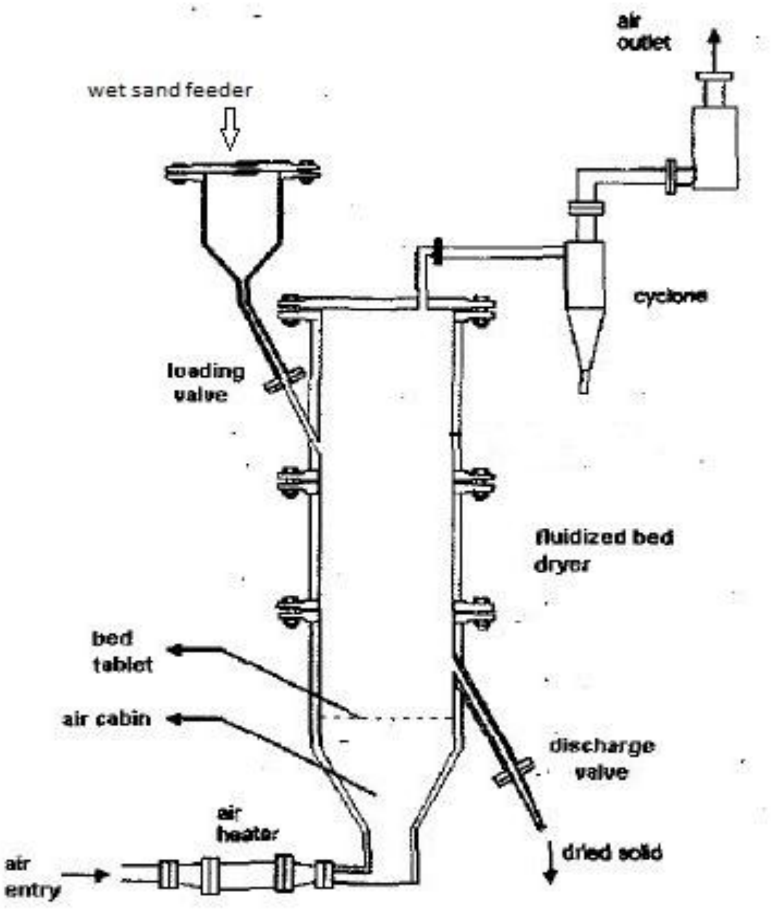


Figure.1: Drying process of sand particles using Fluidized Bed Dryer

Computational Fluid Dynamics (CFD) is becoming an important engineering tool to predict the flow in various types of large industrial equipments. CFD provides both qualitative and quantitative prediction of fluid flows by means of mathematical modelling. CFD enables scientists and engineers to perform numerical experiments (i.e., computer simulations) in a virtual flow laboratory. CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using experimental techniques.

Although the tools for applying single-phase CFD are widely available, application of multiphase CFD is however, still complicated from both the physical and numerical point of view. Furthermore, multiphase CFD simulations are time consuming and therefore average flow predictions in large scale equipment is not obtained readily. Currently there are two approaches for multiphase modelling; the Euler-Lagrange approach and Euler-Euler approach. For gas-solid flow modelling usually Eulerian-Eulerian models also called Granular Flow Models (GFM) are employed. They are more suitable for simulating larger and complex industrial fluidized-bed dryers containing billions of solid particles.

Research work on fluidized bed dryers are improving now a days, because of its applications in large scale industries and its advantages over other drying processes. Chejne and Hernandez,[1] developed a one dimensional steady state mathematical model to simulate the coal gasification process in fluidized bed dryer. The model developed, incorporates two phases namely solid and liquid. The model predicts the temperature and particle size distribution for solid phase and that for the gaseous phase. Izadifar and Mowla,[2] developed a mathematical model to simulate the drying of moist paddy in a fluidized bed dryer. The model proposed was capable of studying the variation of moisture content of paddy through the dryer. The predicted model showed good agreement with the experimental results.

Gareth et. al,[3] presented a paper on the application of chaos analysis to pressure fluctuations in a conical fluidized bed of dry and wet pharmaceutical granules, as it progresses to the dry state. Changes in bed hydrodynamics arising from particle size distribution have been found to be most easily resolvable at low gas velocities, mostly less than 2 m/s.

Ozbey and Soymelez,[4] experimentally investigated the batch drying of wheat grains in a fluidized bed dryer having a swirling flow field in its drying medium. The effect of the air temperature and mass flow rate on the drying process was also studied.

Liang et. al,[5] developed a new numerical model based on two fluid model including the kinetic theory of granular flow, to simulate the coal gasification in a bubbling fluidized bed gasifier. They predicted the flow behaviour of gas and solid phases in the bed and freeboard, which are difficult to measure through the experiments. The relationship between gas composition profiles with the height of gasifier and the contours of temperature, velocity and solid volume fraction were discussed for both phases. Assari et. al,[6] introduced a mathematical model for batch drying based on Eulerian two fluid models. The effects of parameters such as inlet gas temperature and velocity on the moisture content and the temperature of solid and gas at the outlet are discussed.

Rozainee et. al,[7] utilized CFD modelling to determine the trajectories and residence time of burning rice husk particles in the fluidized bed combustor at different secondary air flow rates. From their study, the secondary air flow rate which is 80% of the primary air flow rate, gave the highest average bed temperature and lowest residual carbon content in the ash. Mahmood et. al,[8] studied the hydrodynamics of a vertically vibrating fluidized bed using an Eulerian-Eulerian two fluid model (TFM) incorporating the kinetic theory of granular flow and including the frictional stress effects. They studied the influence of frictional stresses, vibration amplitudes and frequency, on behavior of the particles. The ability of the two-fluid model for predicting the hydrodynamics of vibrating fluidized beds was also discussed.

Jungkee and Hamid,[9] carried out the CFD simulation of a pharmaceutical bubbling bed drying process at three different scales. They studied the gas–solid flow patterns, mixing and drying of pharmaceutical particles for three different scales of bubbling fluidized bed dryers. Their results were compared with the experimental data obtained at the laboratory scale to validate.

Yuping et.al,[10] done their simulations for biomass drying using a novel fluidized bed dryer. Biomass drying is performed mainly in rotary dryers, which occupy a large floor area. To explore the efficient drying of biomass, a fluidized bed dryer was proposed. A high drying rate could be achieved because of the fast mass and heat transfer rate in the fluidized bed dryer.

Askarishahi et.al,[11] in their paper done CFD study on solids flow pattern and solids mixing characteristics in bubbling fluidized bed to find the effect of fluidization velocity and bed aspect ratio on drying process. In their work, using the two-fluid model based on the kinetic theory of granular flow, a set of governing equations was solved by applying finite volume method in 2-D to save the time for simulations. Adam et.al,[12] developed a CFD model of coal gasification in a circulating fluidized bed reactor. Eulerian‑Lagrangian approach was used to simulate the fluidized bed hydrodynamics. Gasification with air and air/steam mixture was considered for their study. Results of the simulations coincide well with the measured syngas composition.

In the present work, CFD simulations are carried out to study the sand particles drying in a fluidized bed dryer. Here the wet sand particles are fed to the dryer through a conveyor. The hot air from the bottom of the dryer passes through the nozzles embedded in the dryer bed and rises up. This upwind do not allow the wet sand particles to settle down. Instead, the sand particles are in a fluidized state inside the dryer. During the fluidized state movement of sand particles from inlet to outlet, the particle drying will occur. This process is simulated with the aid of ANSYS FLUENT 14.5 software.

MATHEMATICAL MODELLING

Governing Equations

The governing equations for Eulerian multiphase model include conservation of mass, momentum and energy equations. Equations of solid and gas phases are developed based on Eulerian-Eulerian model, using the averaging approach. The system of governing equations is summarized below:

Continuity equation for gas phase:

$$\frac{\partial}{\partial t}(\varepsilon_g \rho_g) + \nabla.(\varepsilon_g \rho_g \overrightarrow{v_g}) = 0 \qquad \dots (1)$$

Continuity equation for solid phase:

$$\frac{\partial}{\partial t}(\varepsilon_s \rho_s) + \nabla.(\varepsilon_s \rho_s \overrightarrow{v_s}) = 0 \qquad \dots (2)$$

Conservation of momentum:

Gas phase:

$$\begin{aligned} &\frac{\partial}{\partial t}(\varepsilon_g \rho_g \overrightarrow{v_g}) + \nabla.(\varepsilon_g \rho_g \overrightarrow{v_g v_g}) \\ &\qquad = -\varepsilon_g \nabla p + \nabla.\overline{\overline{\tau_g}} + \varepsilon_g \rho_g g \\ &\qquad + K_{gs}(\vec{v}_g - \vec{v}_s) \qquad \dots (3) \end{aligned}$$

Solid phase:

$$\begin{aligned} &\frac{\partial}{\partial t}(\varepsilon_s \rho_s \overrightarrow{v_s}) + \nabla.(\varepsilon_s \rho_s \overrightarrow{v_s v_s}) \\ &\qquad = -\varepsilon_s \nabla p - \nabla ps + \nabla.\overline{\overline{\tau_s}} + \varepsilon_s \rho_s g \\ &\qquad + K_{sg}(\vec{v}_s - \vec{v}_g) \qquad \dots (4) \end{aligned}$$

Conservation of thermal energy equation:

Gas phase:

$$\begin{aligned} &\frac{\partial}{\partial t}(\varepsilon_g \rho_g h_g) + \nabla.(\varepsilon_g \rho_g \overrightarrow{v_g} h_g) = -\varepsilon_g \frac{\partial p_g}{\partial t} + \overline{\overline{\tau_g}} \\ &\qquad : \nabla \overrightarrow{v_g} - \nabla.\overrightarrow{q_g} + Q_{sg} \\ &\qquad + \dot{m}\Delta H_{vap} \qquad \dots (5) \end{aligned}$$

Solid phase:

$$\begin{aligned} &\frac{\partial}{\partial t}(\varepsilon_s \rho_s h_s) + \nabla.(\varepsilon_s \rho_s \overrightarrow{u_s} h_s) = -\varepsilon_s \frac{\partial p_s}{\partial t} + \overline{\overline{\tau_s}} \\ &\qquad : \nabla \overrightarrow{u_s} - \nabla.\overrightarrow{q_s} + Q_{gs} \\ &\qquad - \dot{m}\Delta H_{vap} \qquad \dots (6) \end{aligned}$$

where ρ is the density, ε is the volume fraction, v is the velocity vector, p is the pressure, τ is the shear stress tensor, g is the gravitational acceleration vector, h is the heat transfer coefficient, q is the heat flux, Q is the heat transfer rate, H is the latent heat and *K* is the drag coefficient.

Geometry and mesh

The first step in CFD simulation of fluidized bed dryer is pre-processing, which has been done using SOLIDWORKS and ICEM CFD software. The initial stage is to model the geometry (Fig.2) for meshing. It is done with SOLIDWORKS software.

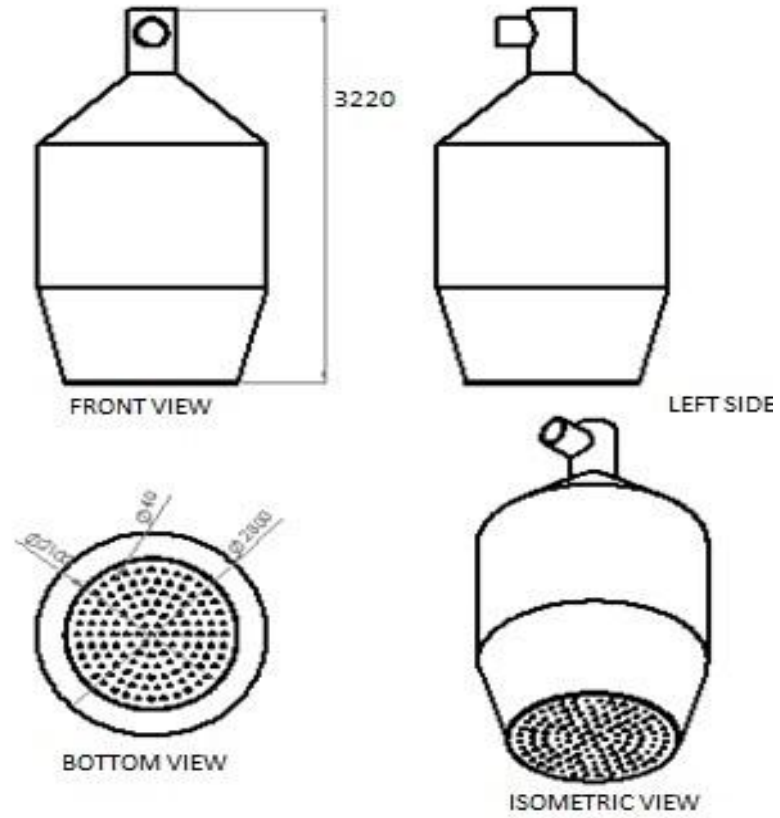


Figure.2: Solidworks model with dimensions in mm

FLUENT needs the domain in which the flow takes place to evaluate the solution. The flow domain as well as the grid generation throughout the domain have been created in ICEM CFD software. The three dimensional geometry created using SOLIDWORKS is imported into ICEM CFD for grid generation. Tetrahedral meshing is used for the current geometry (Fig.3). Three different mesh sizes are generated to study the grid independency of the meshed model. After meshing the necessary initial and boundary conditions are to be imposed in FLUENT 14.5 for starting the simulation.

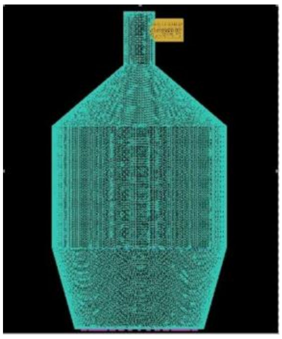


Figure.3: Tetra mesh with element size 65,35,91

NUMERICAL METHEDOLOGY

Boundary and initial conditions

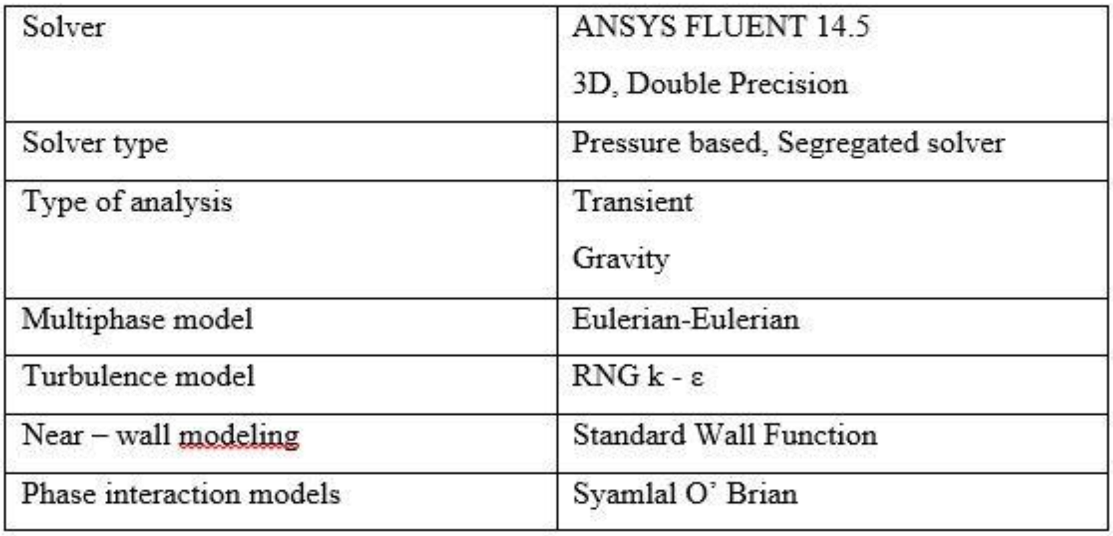
In order to obtain a result with minimal error, appropriate boundary conditions for the computational domain have to be implemented. Here the inlet boundary condition is imposed with velocity inlet and the outlet boundary condition is the pressure outlet, which is set as default pressure. Wall boundary conditions are no-slip condition for both solid and gas phase. The higher viscous effect and higher velocity gradient near the wall have been dealt with the standard wall function. For patching sand volume fraction, the sand in the part of the column up to which the sand particles were initially fed has been used. Initially a sand volume fraction of 0.8 of the static bed height of column has been used and the volume fraction of air at the inlet is set as default value. Table.1 shows the boundary and initial conditions for the CFD analysis of drying sand particles.

Table 1: Boundary and initial conditions

Primary phase	Air
Secondary phase	Sand(granular) Density(2500 kg/m^3) Viscosity(1x10^{-3} kg/m s) Particle size(180 micron)
Mesh type	Tetra mesh
Inlet air temperature	400 K
Inlet air velocity	60 m/s
Outlet air temperature	353 K
Outlet air velocity	85 m/s
Time step size	0.01
Convergence criteria	1x10^{-5}

Table.2 shows the numerical schemes and turbulence model employed in CFD.

Table 2: Solver setup



Solver	ANSYS FLUENT 14.5 3D, Double Precision
Solver type	Pressure based, Segregated solver
Type of analysis	Transient Gravity
Multiphase model	Eulerian-Eulerian
Turbulence model	RNG k - ε
Near – wall modeling	Standard Wall Function
Phase interaction models	Syamlal O' Brian

Grid independency study

For grid independence test, three different meshes (Table.3) are created for simulating the drying process of sand particles.

Table 3: Grid Independence study

Sl. No.	MESH COUNT	EXPERIMENTAL OUTLET AIR VELOCITY [m/s]	SIMULATION OUTLET AIR VELOCITY [m/s]	% ERROR
1	46,18,423	85	92	8.23
2	65,35,919	85	90	5.88
3	87,96,534	85	89	4.7

Here the outlet air velocity is considered as the parameter for grid independence study. Figures 4-6 demonstrates the convergence history of outlet air velocity for three different grid sizes. The actual experimental result at KMML for outlet air velocity is 85 m/s. For a grid size of 4.6 million, the simulation result obtained was 92 m/s. For 6.5 million and 8.7 million mesh, the outlet air velocity obtained are 90 m/s and 89 m/s respectively. This implies that 6.5 million mesh and 8.7 million mesh results provide less error percentage than 4.6 million mesh. As the error percentage between 6.5 million mesh and 8.7 million mesh is minimum and considering the computational time, 6.5 million mesh is considered for all simulations.

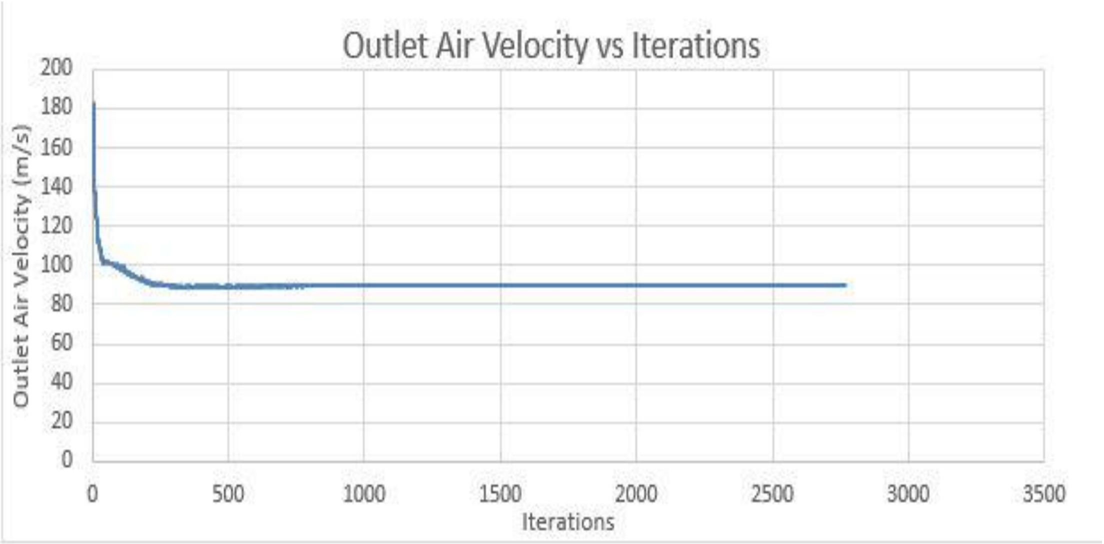


Figure.4: Convergence history of Outlet air velocity for 4.6 million mesh

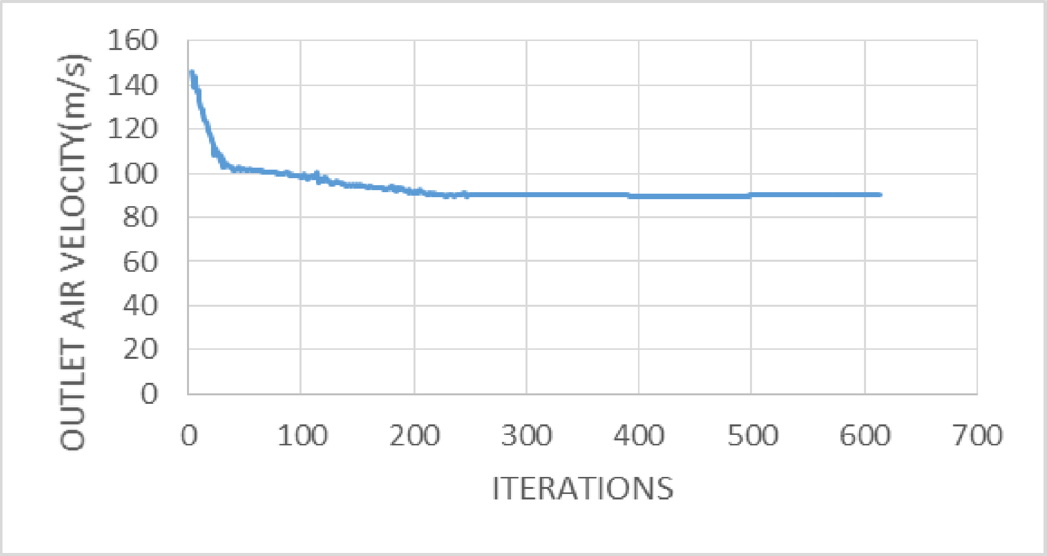


Figure.5: Convergence history of Outlet air velocity for 6.5 million mesh

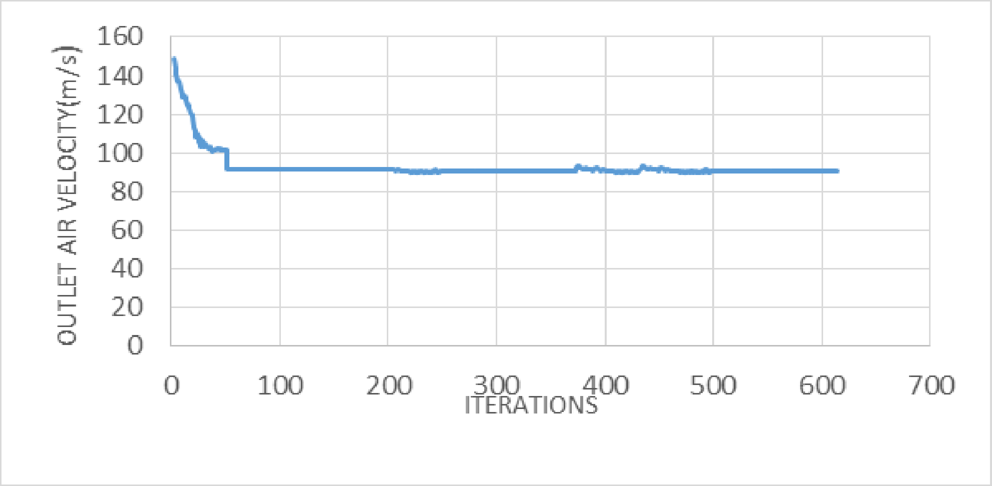
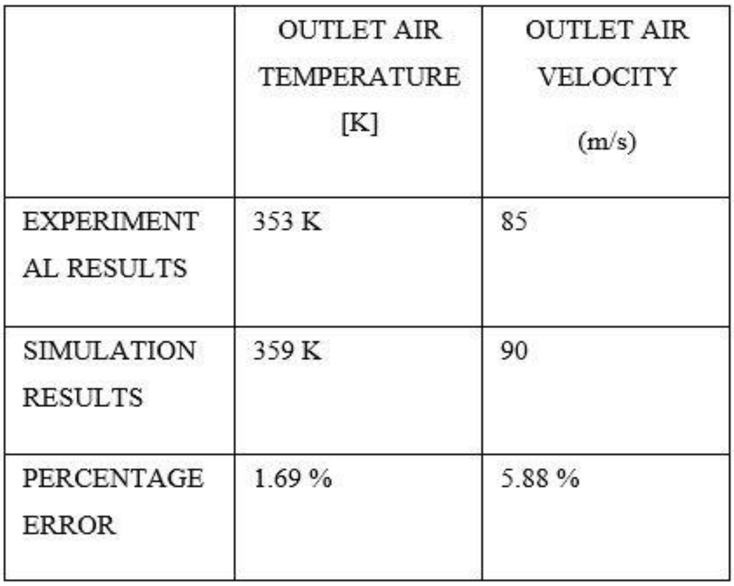


Figure.6: Convergence history of Outlet air velocity for 8.7 million mesh

Validation study

Table.4 shows the validation results. The outlet air temperature in the fluidized bed dryer is monitored experimentally with the PLC circuits and is constant at 353K. Through numerical simulation it is found to be 359K, with a percentage error of 1.69%. Similarly, the percentage error in outlet air velocity is found to be 5.88 %.

Table 4: Validation study



	OUTLET AIR TEMPERATURE [K]	OUTLET AIR VELOCITY (m/s)
EXPERIMENTAL RESULTS	353 K	85
SIMULATION RESULTS	359 K	90
PERCENTAGE ERROR	1.69 %	5.88 %

RESULTS AND DISCUSSION

The simulations are started from time, t = 0 s, until a steady state is obtained for an inlet air temperature of 400 K. Considering the mesh size, three dimensional geometry and multiphase simulation, this study is carried out only for 2.5 seconds. In figures 7-12 the contours of sand volume fraction inside the dryer are plotted for different time.

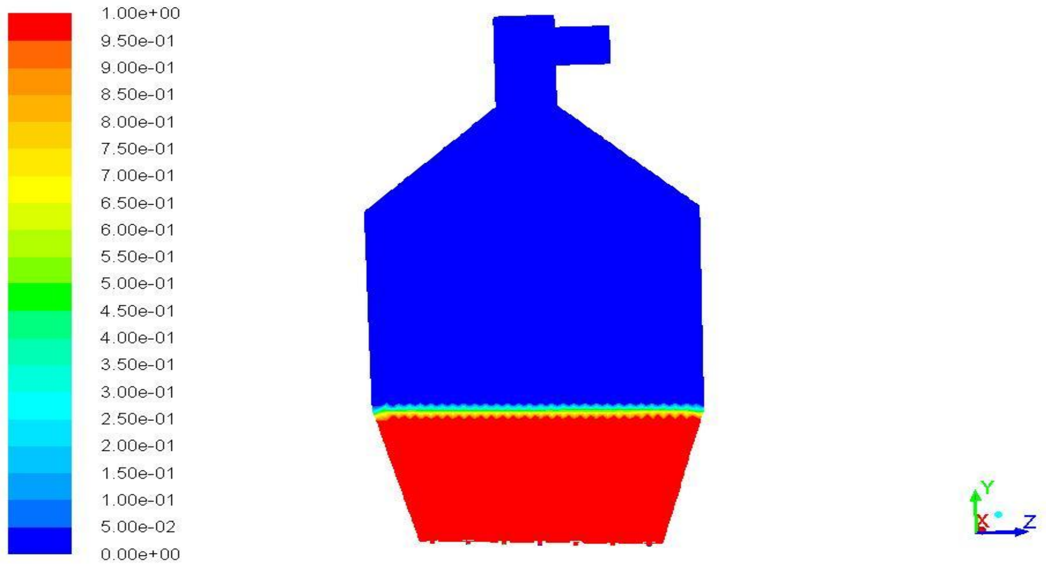


Figure 7: Volume fraction of sand at 0.0s

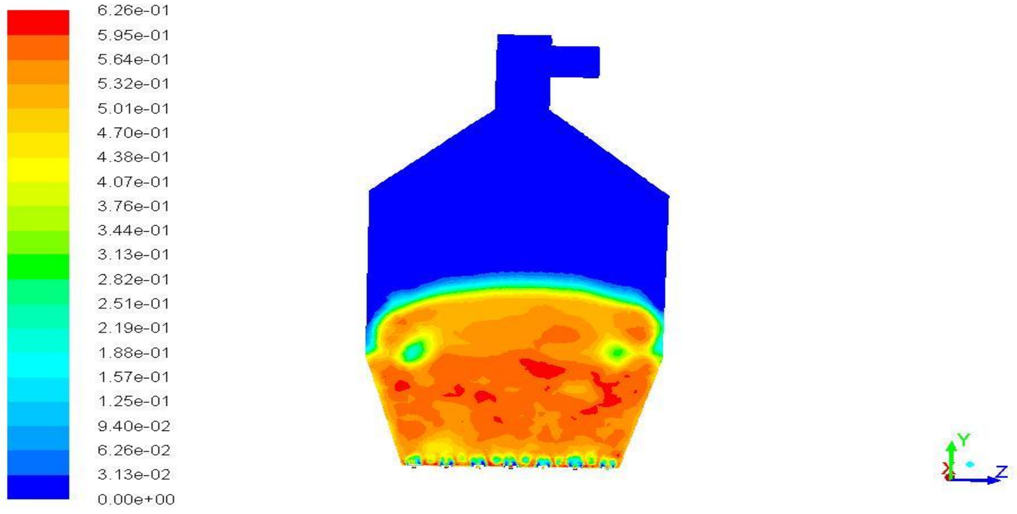


Figure 8: Volume fraction of sand at 0.41s

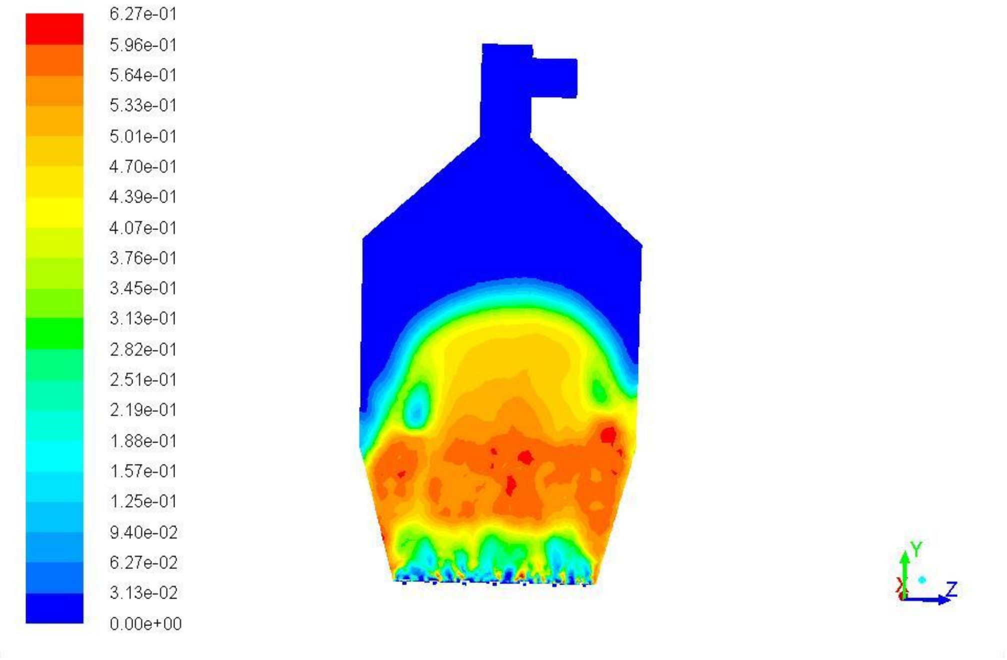


Figure 9: Volume fraction of sand at 0.89s

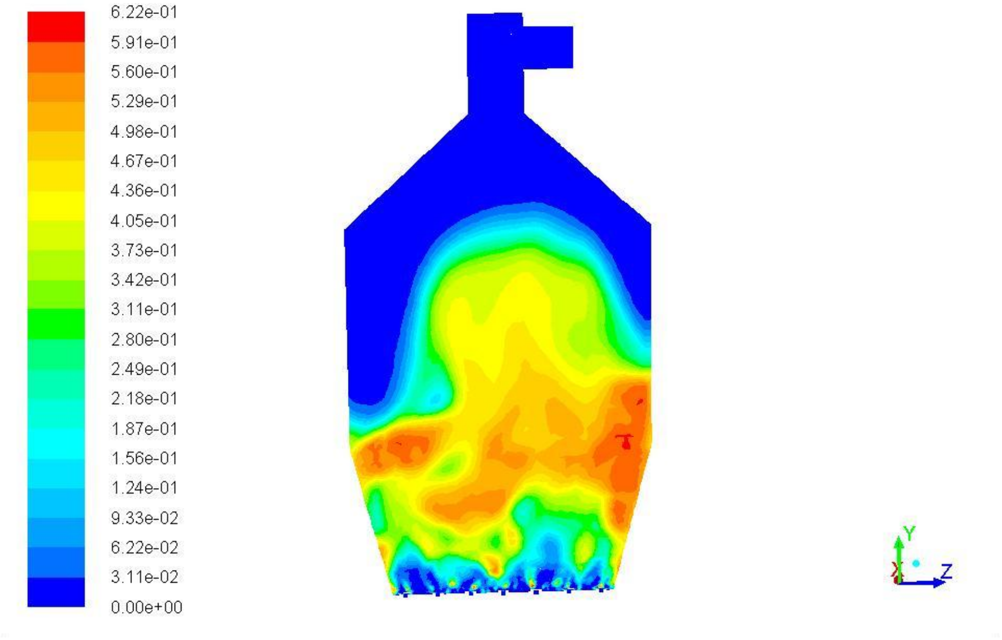


Figure 10: Volume fraction of sand at 1.4s

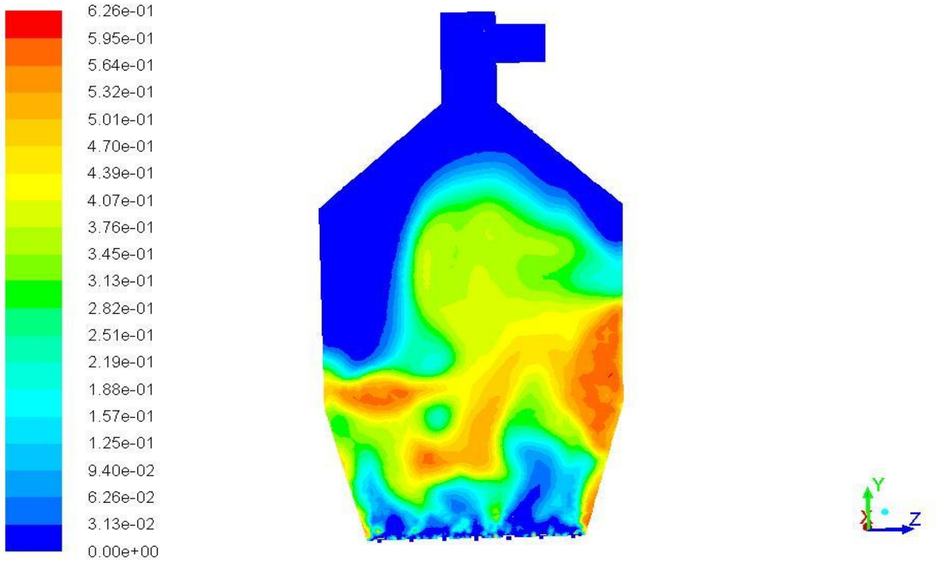


Figure 11: Volume fraction of sand at 1.83s

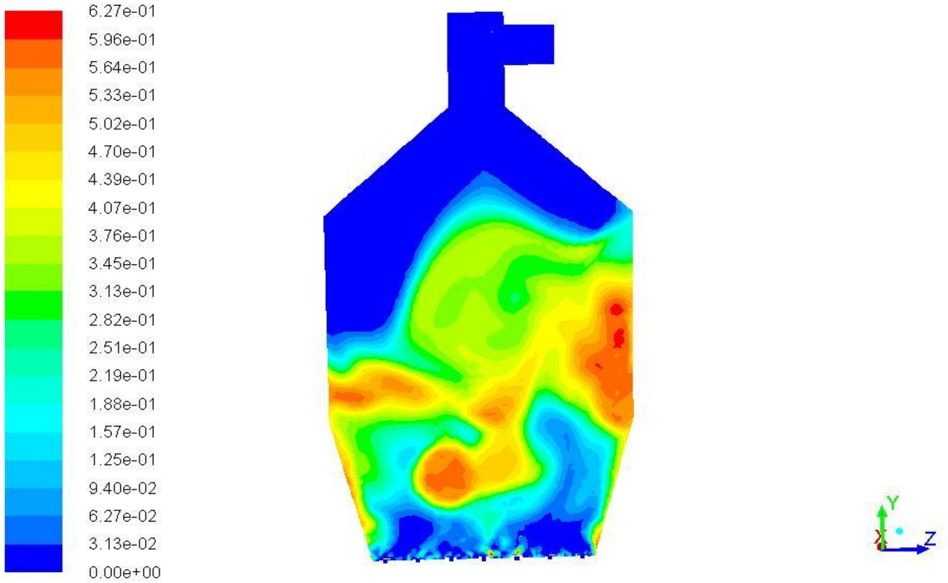


Figure 12: Volume fraction of sand at 2.3s

The fluidization process is nicely captured through this simulation. Initially the highest concentration of sand is at the bottom of the dryer. With the increase in velocity of the upward hot air, the sand particles will start to rise up in the dryer and the sand particles will be in a fluidized state thereafter.

Figures 13 and 14 shows the contours of air and sand temperature distribution inside the dryer. The temperature of air is maximum at the nozzle inlet. Through the nozzles the hot air rises up. This upwind suspends the wet sand particles in a fluidized state. The hot air surrounding the wet sand causes the sand to dry. Thus the temperature of air reduces as it flows out through the outlet of the dryer.

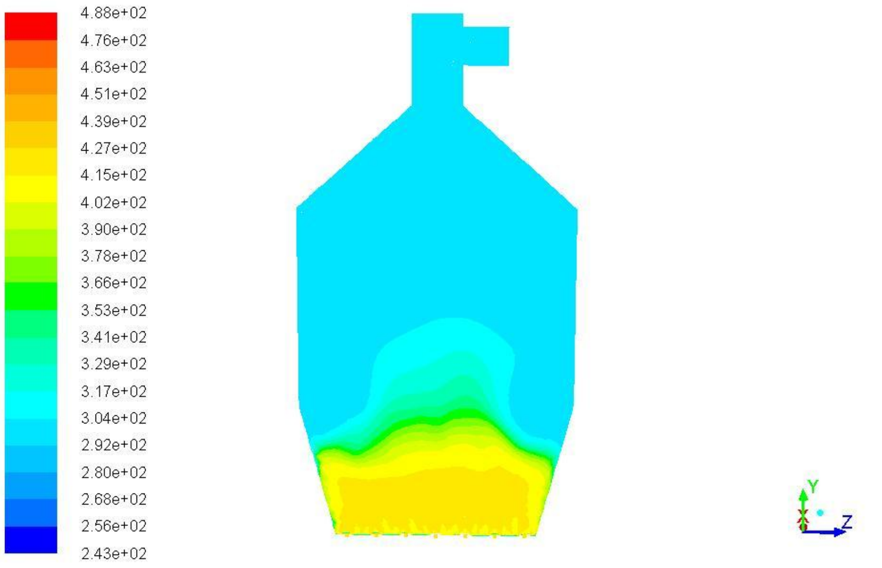


Figure 13: Contour of gas phase (Air) temperature inside dryer

Figure 14: Contour of solid phase (Sand) temperature inside dryer

The temperature of sand particles over time inside the dryer is monitored at different inlet air temperatures as shown in Table.5. It is observed from Fig.15 that with increase in air temperature from 400 K to 440 K, there is increase in sand temperature. In KMML, once the sand temperature is reached 373 K, it is taken out. It is observed that with increase in inlet air temperature, the sand particles obtain 373K much faster showing an increased drying rate of sand. At inlet air temperature, 430 K the change is less compared with other inlet temperatures. So for optimization of dryer the inlet air temperature can be selected as 430 K. Sand temperature at different inlet air temperature is plotted below.

Table 5: Sand temperature at different time seconds inside dryer

TIME [S]	INLET AIR TEMPERATURE				
	400K	410K	420K	430K	440K
0	298	298	298	298	298
0.5	315	318	325	331	333
1	329	335	341	347	354
1.5	346	351	364	371	377
2	358	364	387	396	411
2.5	377	385	402	418	422

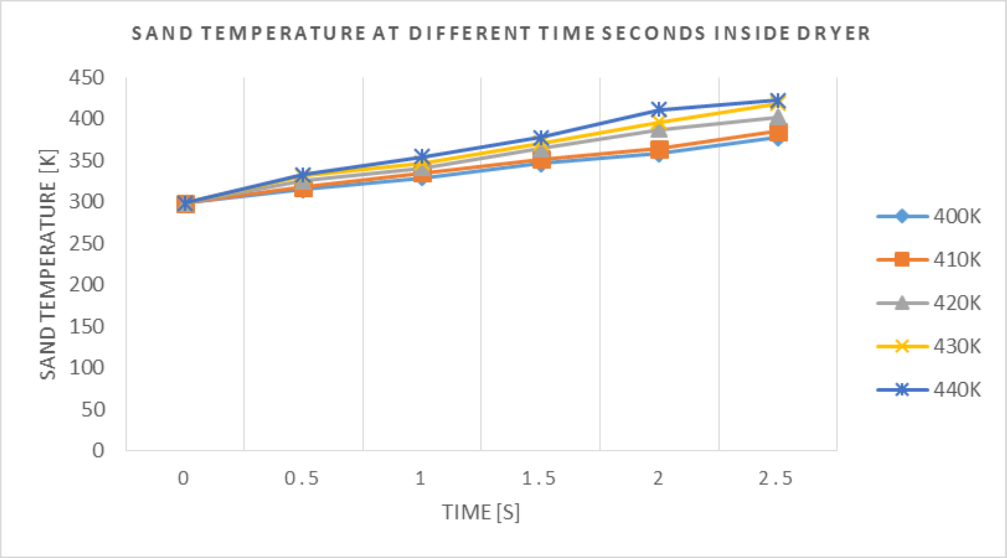


Figure.15: Variation of sand temperature with time for different inlet air temperature

From figure 15, it is observed that as the temperature of the inlet air is increased, the sand particle temperature inside the dryer is also increased. For the inlet temperature of 400K, the sand particles reach 373 K around 2.5 s inside the dryer. With increase in temperature to 410 K, it is reduced to approximately 2.3 s. With increase in inlet air temperature, the sand particles attain 373 K inside the dryer with less time . i.e., the drying rate is increased with increase in temperature. But from 430 K to 440 K the change is less compared with other inlet temperatures. So for the current study, the optimum temperature for inlet air is observed at 430 K.

CONCLUSIONS

CFD study of sand particles drying in a fluidized bed dryer has been carried out using ANSYS FLUENT 14.5. Eulerian multiphase model has been used to simulate the flow of particles inside the dryer. The simulation results are aggregable with the experimental results obtained, for outlet air temperature and outlet air velocity. With the increase in inlet air temperature, the sand particle temperature inside the dryer is also increased. As inlet air temperature is increased, sand particles drying rate is also increased.

REFERENCES

[1] F. Chejne, J.P. Hernandez, “Modelling and simulation of coal gasification process in fluidised bed,” Fuel 81 (2002) 1687-1702.

[2] M. Izadifar, D. Mowla, “Simulation of a cross-flow continuous fluidized bed dryer for paddy rice,” Journal of Food Engineering 58 (2003) 325–329.

[3] Gareth Chaplina, Todd Pugsleya, Conrad Winters, “Application of chaos analysis to pressure fluctuation data from a fluidized bed dryer containing pharmaceutical granule,” Powder Technology 142 (2004) 110– 120.

[4] M. Ozbey, M.S. Soylemez, “Effect of swirling flow on fluidized bed drying of wheat grains,” Energy Conversion and Management 46 (2005) 1495–1512.

[5] Liang Yu, Jing Lu, Xiangping Zhang a, Suojiang Zhang, “Numerical simulation of the bubbling fluidized bed coal gasification by the kinetic theory of granular flow (KTGF),” Fuel 86 (2007) 722–734.

[6] M.R. Assari, H. Basirat Tabrizi, M. Saffar-Avval, “Numerical simulation of fluid bed drying based on two-fluid model and experimental validation,” Applied Thermal Engineering 27 (2007) 422–429.

[7] M. Rozainee, S.P. Ngo, Arshad A. Salema, K.G. Tan, “Computational fluid dynamics modeling of rice husk combustion in a fluidised bed combustor,” Powder Technology 203 (2010) 331–347.

[8] Mahmood Reza Rahimi, Nader Azizi, Seyyed Hossein Hosseini, Goodarz Ahmadi, “CFD study of hydrodynamics behaviour of a vibrating fluidized bed using kinetic-frictional stress model of granular flow,” Korean J. Chem. Eng., 30(3), 761-770 (2013).

[9] Jungkee Jang, Hamid Arastoopour, "CFD simulation of a pharmaceutical bubbling bed drying process at three different scales," Powder Technology 263 (2014) 14–25.

[10] Yuping Liu, Jianghong Peng, Yasuki Kansha, Masanori Ishizuka, Atsushi Tsutsumi, Dening Jia, Xiaotao T. Bi, C.J. Limb, Shahab Sokhansanj, "Novel fluidized bed dryer for biomass drying," Fuel Processing Technology 122 (2014) 170–175.

[11] Maryam Askarishahi, Mohammad-Sadegh Salehi, Hamid Reza Godini, GunterWozny, "CFD study on solids flow pattern and solids mixing characteristics in bubbling fluidized bed: Effect of fluidization velocity and bed aspect ratio," Powder Technology 274 (2015) 379–392.

[12] Adam Klimanek, Wojciech Adamczyk, Anna Katelbach-Woz niak, Gabriel Wecel, Andrzej Szlek, "Towards a hybrid Eulerian–Lagrangian CFD modelling of coal gasification in a circulating fluidized bed reactor," Fuel 152 (2015) 131–137.